© 2017 IJSRSET | Volume 3 | Issue 3 | Print ISSN: 2395-1990 | Online ISSN : 2394-4099
Themed Section: Engineering and Technology

The Comparative Study on Analysis Results of Multi-Storeyed Commercial Building (G+12) By Staad. Pro and Etabs

Shaik Kamaluddhin[1], A. B. S. Dadapeer[2]

*[1]Department of Civil Engineering, St.mark Educational Institution Society Group of Institutions, Anantapur, Andhra Pradesh, India
*[2]Asst. Prof, Department of Civil Engineering, St.mark Educational Institution Society Group of Institutions, Anantapur, Andhra Pradesh, India.

ABSTRACT

Structural Analysis is a branch which involves in the determination of behaviour of structures in order to predict the responses of real structures such as buildings, bridges, trusses etc. Under the improvement of expected loading & external environment during the service life of structure. The results of analysis are used to verify the structure fitness for use. Computer software's are also being used for the calculation of forces, bending moment, stress, strain &deformation or deflection for a complex structural system. The principle objective of this project is "The Comparative Study on Analysis Results of Multi-storeyed Commercial Building (G+12) by STAAD.Pro and ETABS". STAAD.Pro is one of the leading software's for the design of structures. In this project I had analysed the G+12 building for finding the shear forces, bending moments, deflections details for the structural components of building (such as Beams, Columns & Slabs) to develop the economic design. ETABS is also leading design software in present days used by many structural designers. Here I have also analysed the same structure using ETABS software for the design. Finally I made an attempt to define the economical section of G+12 multi-storeyed building using STAAD.Pro and ETABS comparatively.

Keywords: Shear Force, Bending Moment, Deflection, Structural Analysis, Etabs, Staad Pro

I. INTRODUCTION

An RCC framed structure is basically an assembly of slabs, beams, columns and foundation inter-connected to each other as a unit. The load transfer, in such a structure takes place from the slabs to the beams, from the beams to the columns and then to the lower columns and finally to the foundation which in turn transfers it to the soil. The floor area of a R.C.C framed structure building is 10 to 12 percent more than that of a load bearing walled building. Monolithic construction is possible with R.C.C framed structures and they can resist vibrations, earthquakes and shocks more effectively than load bearing walled buildings. Speed of construction for RCC framed structures is more rapid. Reinforced concrete is a composite material in which concrete's relatively low tensile strength and ductility are counteracted by the inclusion of reinforcement having higher tensile strength and ductility.

II. DESIGN LATERAL FORCE

The procedure recommended for the determination of lateral force in IS: 1893-2002(Part 1) performing are based on the approximation that effects of yielding can be accounted for by linear analysis of the building using design spectrum. This analysis is carried out by either equivalent lateral force procedure or dynamic analysis procedure given in the clause 7.8 of IS: 1893-2002 (Part 1). The main difference between the two procedures lies in the magnitude and distribution of lateral forces over the height of the building. In the equivalent lateral force procedure the magnitude of forces is based on an estimation of the fundamental period and on the distribution of forces as given by a simple empirical formula that is appropriate only for regular buildings. The following sections will discuss in detail the above mentioned procedures of seismic analysis.

III. EQUIVALENT LATERAL FORCE METHOD

The total design lateral force or design base shear along any principal direction is given in terms of design horizontal seismic coefficient and seismic weight of the structure. Design horizontal seismic coefficient depends on the zone factor of the site, importance of the structure, response reduction factor of the lateral load resisting elements and the fundamental period of the structure. The procedure generally used for the equivalent static analysis is explained below:

Determination of fundamental natural
Period (Ta) of the buildings:

Ta = 0.075h0.75Moment resisting RC frame building without brick infill wall Ta = 0.085h 0.75Moment resisting steel frame building without brick infill walls

Where, h -is the height of building in m

d - is the base of building at plinth level in m, along the considered direction of lateral force.

Determination of base shear (VB) of the building:

VB =Ah×W

Where, Ah = is the design horizontal seismic coefficient, which depends on the seismic zone factor (Z), importance factor (I), response reduction factor (R) and the average response acceleration coefficients (Sa/g). Sa/g in turn depends on the nature of foundation soil (rock, medium or soft soil sites), natural period and the damping of the structure.

IV. RESPONSE SPECTRUM METHOD:

The response spectrum technique is really a simplified special case of modal analysis. The modes of vibration are determined in period and shape in the usual way and the maximum response magnitudes corresponding to each mode are found by reference to a response spectrum. The response spectrum method has the great virtues of speed and cheapness. There are two major disadvantages of using this approach. First, the method produces a large amount of output information that can require an enormous amount of computational effort to conduct all possible design checks as a function of time. Second, the analysis must be repeated for several different earthquake motions in order to assure that all the significant modes are excited, since a response spectrum for one earthquake, in a specified direction, is not a smooth function.

Loadings:

This stage involves determination of various types that are acting on the structures. The values of types of loads are taken from the relevant IS-codes.

Types of Loads:

Various types of loads on a structure and requiring consideration in design are

Dead Load:

This is the permanent of stationary load like self-weight of structural elements. This includes

a) self-weight b)weight of finished c)weights of partitions walls etc..,

Live Load: (As Per Is-875-Part-2)

These are non-permanent or moving loads. Imposed loads (fixed) weight of fixed beams in auditoriums. Fixed machinery, partition walls. These loads though, fixed in positions cannot relied up on to act permanently throughout the life of the structure.

Imposed loads (not fixed) these loads can change either in magnitude or position very often such as traffic loads weight of furniture etc.

Dead Loads:

R.C.C	25 kN/m3
P.C.C	24 kN/m3
Brick masonry	22 kN/m3
Floor finishes	0.7 kN/m3

Live Loads:

On floors	4 kN /m2
On roofs	2 kN /m2

V. ANALYSIS AND DESIGN OF G+12 BUILDING USING STAAD. Pro

Step - 1: Creation of nodal points.

Based on the column positioning of plan we entered the node points into the

STAAD file

Step - 2: Representation of beams and columns. By using add beam command we had drawn the beams and columns between the corresponding node points.

Step - 3: 3D view of structure. Here we have used the Transitional repeat command in Y direction to get the 3D view of structure.

Step - 4: Supports and property assigning.

After the creation of structure the supports at the base of structure are specified as fixed. Also the materials were specified and cross section of beams and columns members was assigned.

Step - 5: 3D rendering view. After assigning the property the 3d rendering view of the structure can be shown

Step - 6: Assigning of seismic loads. In order to assign Seismic loads firstly we have defined the seismic loads according to the code IS 1893:2002 with proper floor weights. Loads are added in load case details in +X,-X, +Z,-Z directions with specified seismic factor.

Step - 7: Assigning of wind loads. Wind loads are defined as per IS 875 PART 3 based on intensity calculated and exposure factor. Then loads are added in load case details in +X,-X, +Z,-Z directions.

Step - 8: Assigning of dead loads. Dead loads are calculated as per IS 875 PART 1 for external walls, internal walls, parapet wall including self-weight of structure.

Step - 9: Assigning of live loads. Live loads are assigned for every floor as 4 kN/m2 based on IS 875 PART 2.

Step - 10: Adding of load combinations.

After assigning all the loads, the load combinations are given with suitable factor of safety as per IS 875 PART 5.

VI. ANALYSIS AND DESIGN OF G + 12 BUILDING USING ETABS:

Step - 1: Step by Step procedure for ETABS Analysis The procedure carried out for modelling and analysing the structure involves the following flow chart.

Step - 2: Creation of Grid points & Generation of structure. After getting opened with ETABS we select a new model and a window appears where we had entered the grid dimensions and story dimensions of our building. Here itself we had generated our. 3D structure by specifying the building details in the following window.

Step - 3: Defining of property. Here we had first defined the material property by selecting define menu material properties. We add new material for our structural components (beams, columns, slabs) by giving the specified details in defining. After that we define section size by selecting frame sections as shown below & added the required section for beams, columns etc.

Step - 4: Assigning of Property After defining the property we draw the structural components using command menu. Draw line for beam for beams and create columns in region for columns by which property assigning is completed for beams and columns.

Step - 5: Assigning of Supports. By keeping the selection at the base of the structure and selecting all the columns we assigned supports by going to assign menu.

Step - 6: Defining of loads. In STAAD program we define only seismic and wind loads whereas in ETABS all the load considerations are first defined and then assigned. The loads in ETABS are defined as using static load cases command in define menu.

Step - 7: Assigning of Dead loads. After defining all the loads, dead loads are assigned for external walls, internal walls.

Step - 8: Assigning of Live loads. Live loads are assigned for the entire structure including floor finishing.

Step - 9: Assigning of wind loads. Wind loads are defined and assigned as per IS 875 1987 PART 3 by giving wind speed and wind angle in X,X1,Z & Z1 directions as 0 , 180 , 90 , 270 respectively.

Step - 10: Assigning of Seismic loads. Seismic loads are defined and assigned as per IS 1893: 2002 by giving zone, soil type, and response reduction factor in X and Y directions.

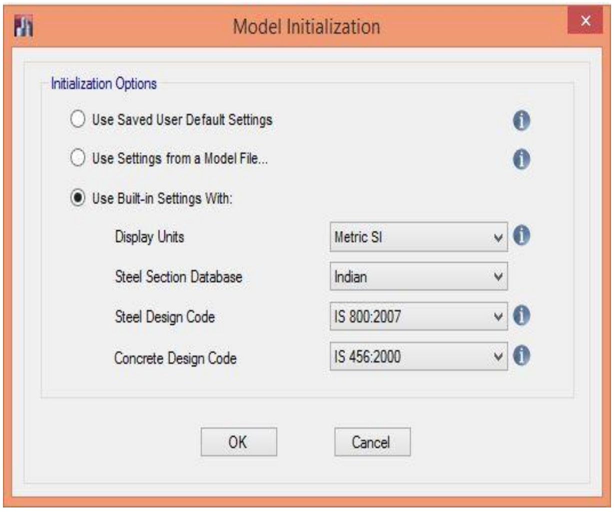
Step - 11: Assigning of load combinations. Load combinations are given as mentioned in STAAD. Pro based on IS 875 1987 PART. 5 using load combinations command in define menu.

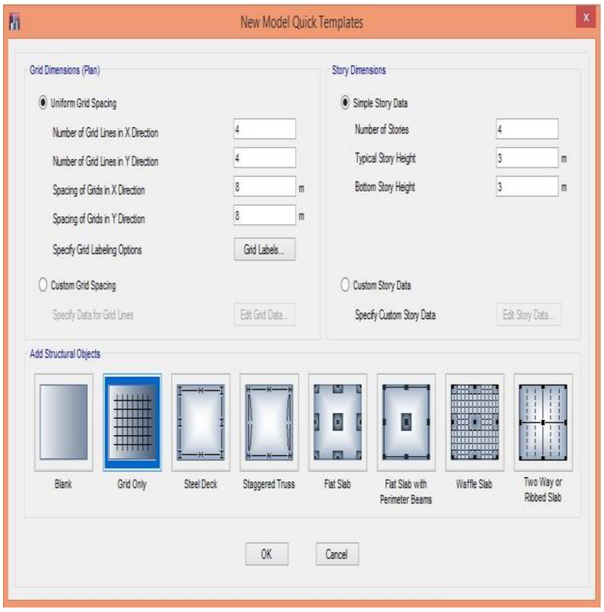
Step - 12: Analysis, after the completion of all the above steps we have performed the analysis and checked for errors.

VI. ETABS MODELLING

DEFINE GEOMETRY:

The Building Plan Grid System and Storey Data form is used to specify horizontal and vertical grid line spacing, storey data, storey elevation and units. They automatically add the structural objects with appropriate properties to the model.

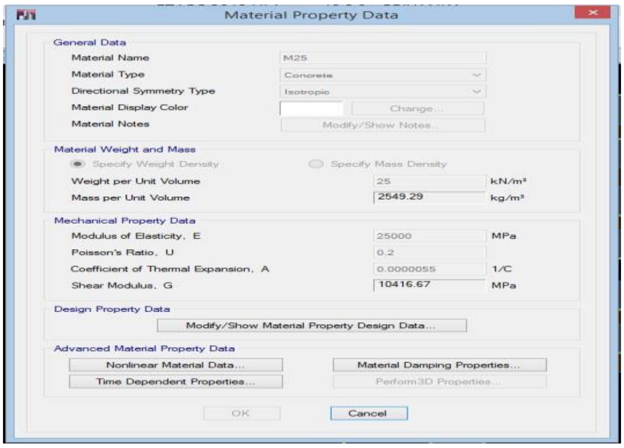


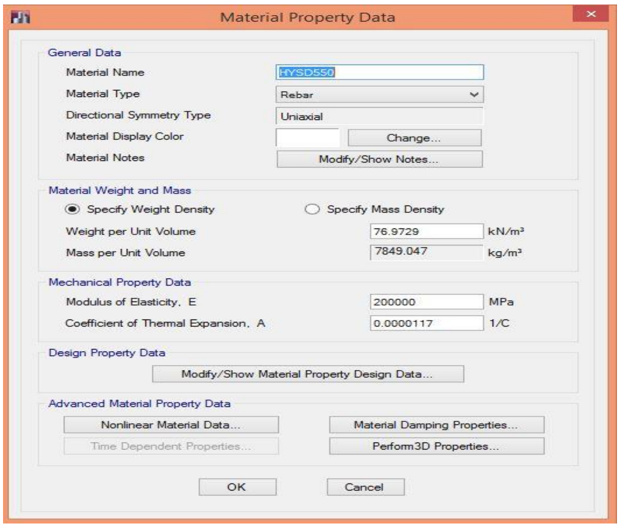


Building Plan Grid System and Storey Data Definition

DEFINE MATERIAL PROPERTY:

The material properties of each object in the model is specified in the appropriate form. The material used is concrete, the grade of concrete, the properties of concrete such as Mass per unit volume, Modulus of Elasticity of concrete, Poisson ratio are specified and for steel yield strength is specified.

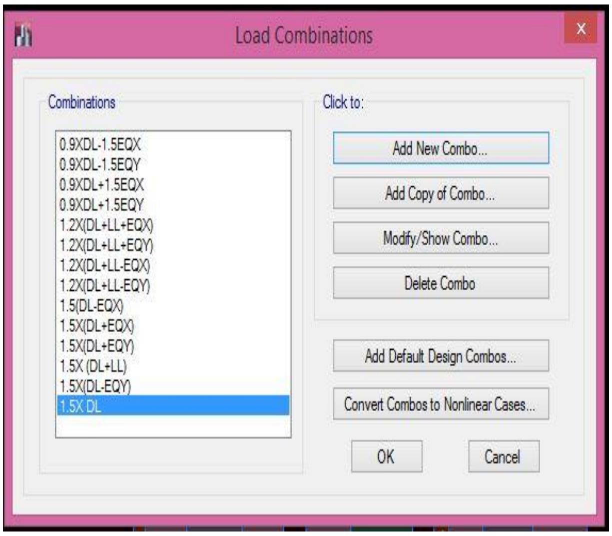




Material property data form.

VII. DEFINE LOAD COMBINATIONS:

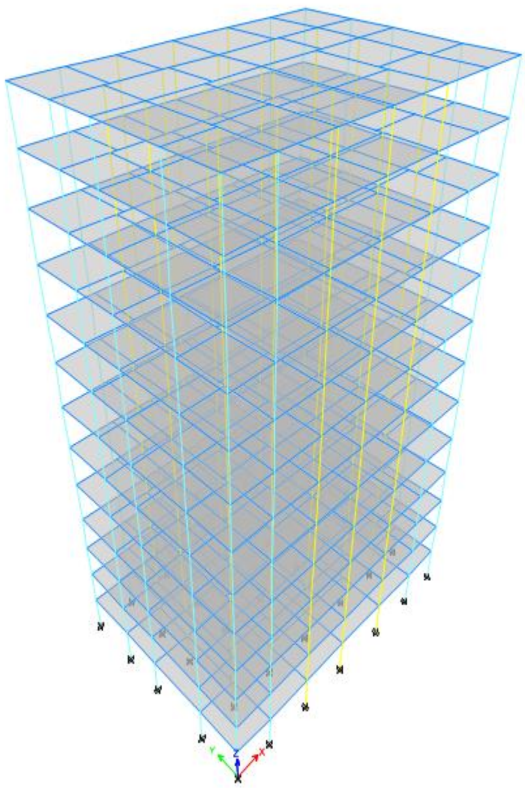
Define the load combinations in the appropriate form; Select the Add new combo option, then enter the load combination name, assign the loads with suitable scale factor. When the combination is defined, it applies to the results for every object in the model.

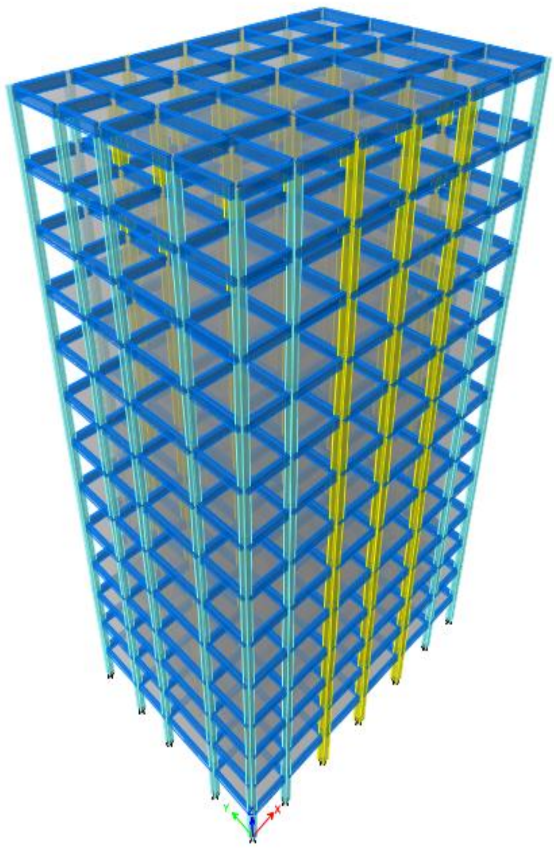


Define load combinations

MODEL OUTPUT:

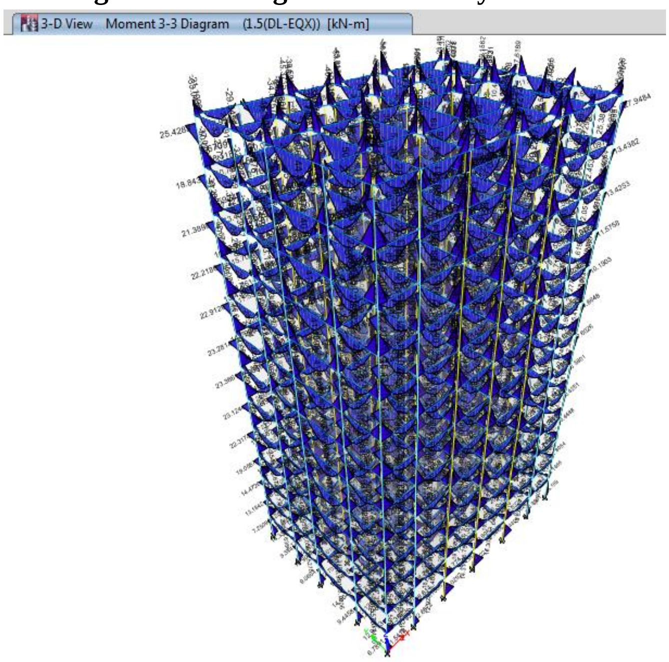
OUTPUT: 3D-MODEL OF A RC FRAME



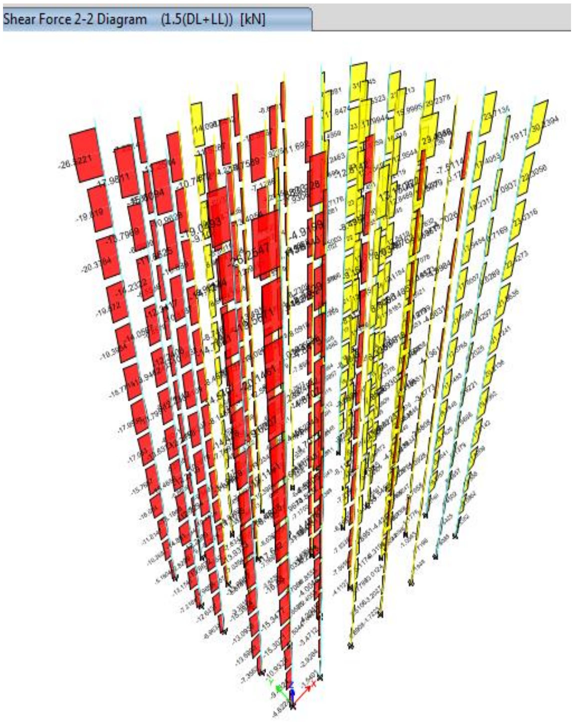


3D model of RC frame

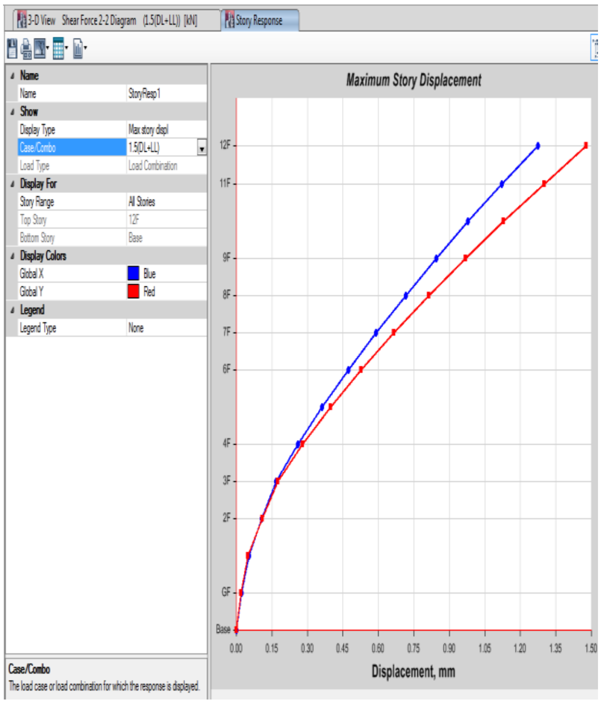
Bending Moment Diagram from Analysis:



Shear Force from Analysis:



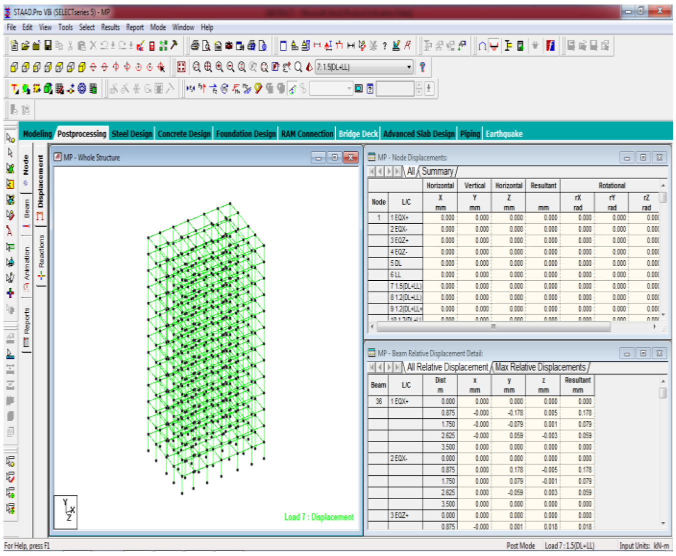
Shear Force from analysis



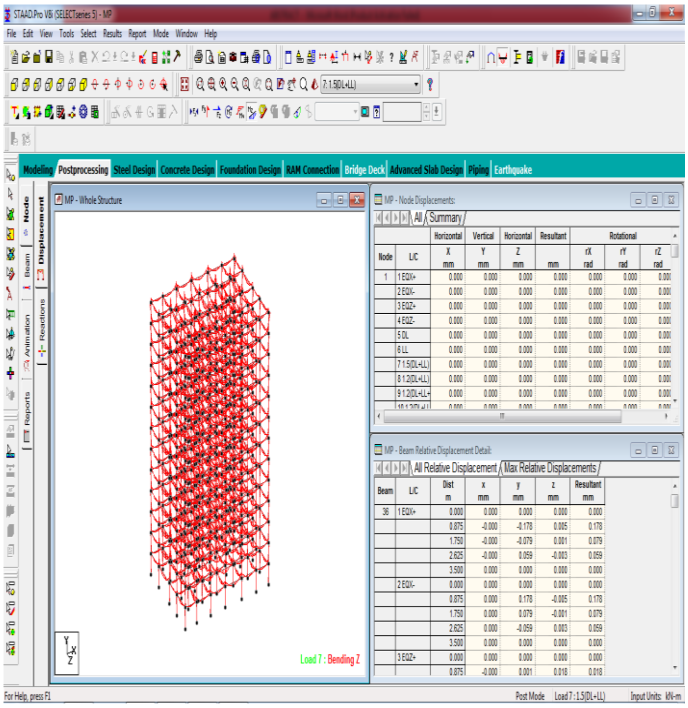
Storey Displacement Curve Under Factored Dead Load+ Live Load

VIII. MODEL OUTPUT

Deflection Diagram from Analysis:



Bending Moment Diagram from Analysis:



IX. CONCLUSION

Actually the results of analysis should match with the classical analysis solutions, whatever the type of analysis we do using whatever S/W package. The finer thing that we should note in all these packages is that, there are parameters/properties such as boundary conditions, material properties etc., which are applied to the generated model by the software package by default at the initial stage. By the analysis results, we can find that the base reactions for the dead load of whole structure is coming little bit different from all the software's. Besides this, we can find base reactions for the live load of the building are equal through both the software's. Furthermore, the bending moment and shear force are also coming nearly same for the considered sample column by all the structural softwares.

X. REFERENCES

[1]. Giordano, M. Guadagnuolo and G. Faella "PUSHOVER ANALYSIS OF PLAN IRREGULAR MASONRY BUILDINGS" The 14th World Conference on Earthquake Engineering October 12-17, 2008, Beijing, China.

[2]. Prof. K S Sable, Er. V A Ghodechor, Prof. S B Kandekar "Comparative Study of Seismic Behavior of Multistorey Flat Slab and Conventional Reinforced Concrete Framed Structures".

[3]. Belejo, R. Bento & C. Bhatt "Comparison of different computer programs to predict the seismic performance of SPEAR building by means of Pushover Analysis the SPEAR building by means of Pushover Analysis" Instituto Superior Técnico, Lisbon, Portugal

[4]. "DISPLACEMENT BASEDDESIGN OF VERTICALLY IRREGULAR FRAMEWALL STRUCTURES"- Rose school thesis work by SUHAIB SALAWDEH.

[5]. "Performance Based Seismic Design and Assessment of Irregular Steel Structures.

[6]. "Seismic design of vertically irregular reinforced concrete structures" - Thesis work submitted by Satrajit Das.

[7]. McCrum Lunch "Seismic Analysis of Braced Plan Irregular Structures Using Hybrid Testing Daniel" lecture at the Department of Civil, Structural & Environmental Engineering, Trinity College Dublin on 30th November 2011

[8]. Shaikh Abdul Aijaj Abdul Rahman, Girish Deshmukh "Seismic Response of Vertically Irregular RC Frame with Stiffness Irregularity at Fourth Floor" ISSN 2250-2459,ISO 9001:2008 Certified Journal, Volume 3, Issue 8, August 2013.